# Design, CFD analysis and numerical optimization of diffuser used in annular type combustion chamber for small gas turbine engine

**Vasant Patel[#1], D.C.Gosai[#2], Dr.K.V.Chaudhari[#3]**

[#1] PG Student, Mechanical Engineering Department, SVMIT-Bharuch,Gujarat Technological University.

[#2] Asso.Prof, Mechanical Engineering Department, SVMIT-Bharuch, Gujarat Technological University.

[#3]Asst.Prof, Mechanical Engineering Department, GEC-Bharuch, Gujarat Technological University.

[1]mevasant73@gmail.com,[2]dip_gosai@yahoo.co.in,[3]kamleshchaudhari@hotmail.com

**Abstract—**

Gas turbines are widely used in aircraft engine as well as in gas based power plant to generate electricity. The combustion chamber used for the combustion of fuel and generate high pressure gas with uniform exit temperature. In gas turbine engine compressor is used to increase the pressure of air when air is coming out of this compressor it is having very high velocity of 150-170 m/s and it is impossible to make the combustion of fuel at such high velocity thus diffuser needs to reduce the velocity of air makes the stable and complete combustion of fuel. Most of the the combustion chamber provide only one inlet at the inlet of the diffuser and let the flow divide by itself into liner and casing .in this present paper design of diffuser is carried out using continuity equation and further simulation is carried out using CFD tool cfx in ANSYS 14.5 to check that desire mass flow rate and velocity of air is achieved at diffuser outlet and snout outlet.

**Keywords—** **Diffuser, Annular type combustion chamber, Design, Numerical optimization, Small gas turbine**

## I. Introduction

In axial-flow compressors the stage pressure rise is very dependent on the axial flow velocity. To achieve the design pressure ratio in the minimum number of stages, a high axial velocity is essential; in many aircraft engines, compressor outlet velocities may reach 150 m/s or higher. It is of course impractical to attempt to burn fuels in air flowing at such high velocities. Thus, before combustion can proceed, the air velocity must be greatly reduced, usually to about one fifth of the compressor outlet velocity. This reduction in velocity is accomplished by fitting a diffuser between the compressor outlet and the upstream end of the liner [7].

In its simplest form, a diffuser is merely a diverging passage in which the flow is decelerated and the reduction in velocity head is converted to a rise in static pressure. The efficiency of this conversion process is of considerable importance because any losses that occur are manifested as a fall in total pressure across the diffuser. In long diffuser of low divergence angle, the pressure loss is high due to skin friction along the walls [7].

## II. DIFFUSER DESIGN

Design of diffuser is carried out for the gas turbine combustor used in gas turbine engine of 15 kw capacity which is operated using CNG as fuel. For the design purpose of diffuser following parameters are used which is calculated through Brayton cycle analysis, energy balance of combustor and combustion chamber dimension.

Table 1: list of parameters required for diffuser design

| | |
|---|---|
| Total Mass flow Rate of air | 0.3377 kg/s |
| Density of air | 2.3 kg/m$^3$ |
| Frontal air mass flow rate | 0.01546 kg/s |
| Combustion chamber centre line distance | 40.57 mm |
| Diffuser inlet velocity [8] | 150 m/s |
| Snout inlet velocity [8] | 50 m/s |
| Divergence angle [2] | 9$^0$ |

For inlet section of diffuser velocity of air is assumed to be 150 m/s [8] and density of air is 2.3 kg/m$^3$ and frontal mass flow rate of air is 0.3377 kg/s so inlet area can be calculated for diffuser section using continuity equation,

$$A_{diffuser} = \frac{W_{fronal\,air}}{(V_{diffuser}\delta_{frontal\,air})}$$

$$A_{diffuser} = 0.000979 \text{ m}^2$$

Taking centre line distance as H=40.57 mm, inner and outer radius at diffuser inlet can be calculated as,

$r_{oinlet}$=42.49 mm

And $r_{iinlet}$ =38.65 mm

Diffuser outlet side inner and outer radius is same as inner and outer radius of casing of combustor as outlet side of diffuser is connected to casing of combustor.so inner and outer radius at diffuser outlet can be given as,

$r_{ooutlet}$= 58.6 mm

And $r_{ioutlet}$ =18.67 mm

**Length of diffuser**

Diffuser length can be calculated for divergence angle of 9 [2] as,

$$\tan 9 = \frac{radius\ of\ outer\ casing - snout\ outer\ radius}{diffuser\ length}$$

Diffuser length= 78.03 mm

## III. SNOUT DESIGN

Design of snout is being carried out with continuity equation, Further, the limiting velocity of air at the exit of snout should preferably be less their 9 m/s [8] and for the air density of 2.3 kg/ $m^3$ and mass flow rate of frontal air 0.01546 kg/s area of snout can be calculated as,

$$A_{snout} = \frac{W_{fronal\,air}}{(V_{snout}\delta_{frontal\,air})}$$

$$=0.0006\ m^2$$

Taking centre line distance as 40.57 mm , inner and outer radius of snout can be calculated as,

$$r_{oinlet} = 41.94\ mm$$

And $r_{iinle\ t} = 39.40$ mm

Snout outlet side inner and outer radius is same as inner and outer radius of liner of combustor as outlet side of snout is connected with liner.so inner and outer radius at snout outlet can be given as,

$$r_{ooutlet} = 27.05\ mm$$

And $r_{ioutlet} = 54.10$ mm

**Snout length**

Snout length can be calculated for divergence angle of 9 [2] as,

$$\tan 9 = \frac{outer\ radius\ of\ diffuser - snout\ outer\ radius}{snout\ length}$$

snout length= 28.93 mm

## IV. CFD ANALYSIS AND NUMERICAL OPTIMIZATION

CFD analysis of diffuser is carried out to check that mass distribution is carried out as per analytical calculation or not and also to check to that desire velocity is achieved at diffuser side and snout side or not. Diffuser geometry is prepared in Ansys 14.5 and its model is shown in figure 1.

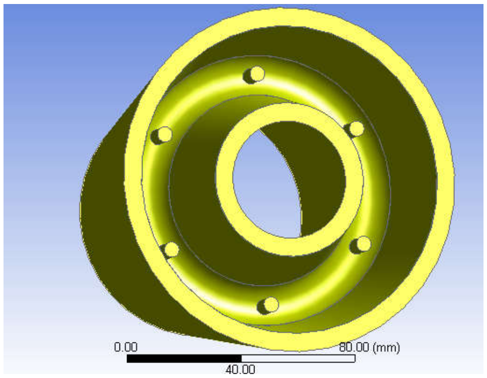


**Figure 1 diffuser model**

**Mesh Generation**

Meshing of diffuser assembly is generated in cfx automatically by providing high smoothing, slow transition and fine span angle centre which gives 83902 nodes and 410225 elements.

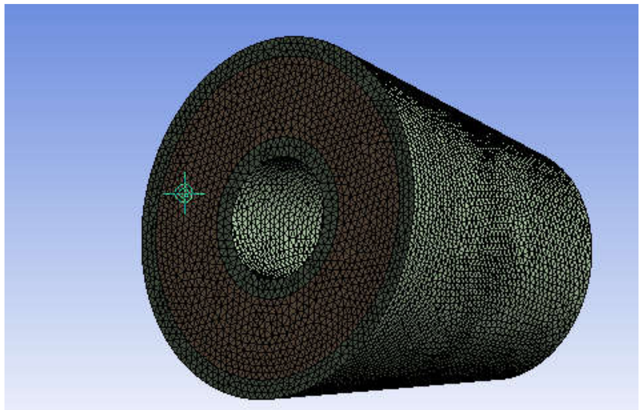

**Figure 2 mesh model of diffuser**

**Numerical models and boundary conditions**
Turbulence Model: k-ω for Steady State Simulations
Air inlet: Specified by mass flow rate =0.3377 kg/s
Outlet: Average Static Pressure Zero
Wall: Adiabatic wall condition

**Boundary Condition**
In analysis of diffuser boundary condition are given at the inlet of diffuser and at the outlet of diffuser and snout.at the inlet of diffuser boundary condition given in the form of mass flow rate of air and it is given as 0.3377 kg/s while at the out let of diffuser and snout average static pressure is zero.

**Result**
Design optimization is carried out by numerical simulation using CFD tool cfx in ANSYS 14.5.when analysis is carried out for preliminary design desire mass flow rate is not achieved, snout outlet is more than required while outer and inner annulus mass flow rate is less than required once so numerical optimization is carried out by modification of design to restrict the flow at snout inlet closed semi-circular rim with holes of diameter 5.34 mm are provided and desire mass flow rate of snout side is achieved but still there is less mass flow in inner annulus hence inner radius is reduced and desire mass flow rate is achieved.

As shown in Table 2 in preliminary design mass flow rate through snout and inner annulus is less than analytical and hence optimization of diffuser is carried out to achieve desire mass flow by changing hole diameter and further it is accomplished by reducing inner radius of diffuser.

Table 2: Summary of the optimization result

| Sr.no | Hole Diameter (mm) | Inner radius (mm) | Outer annulus mass flow rate (kg/s) | Inner annulus mass flow rate (kg/s) | mass flow rate (kg/s) |
|---|---|---|---|---|---|
| 1 | 5.34 | 38.65 | 0.2272 | 0.09897 | 0.01124 |
| 2 | 5.6 | 38.65 | 0.2257 | 0.9925 | 0.01287 |
| 3 | 6.2 | 38.65 | 0.2228 | 0.09900 | 0.01589 |
| 4 | 6.5 | 36 | 0.2003 | 0.1196 | 0.01789 |
| 5 | 6.2 | 34.5 | 0.1874 | 0.1339 | 0.01652 |

The CFD analysis of this diffuser gives the velocity contours and velocity profile at exit section. The gradual flow deceleration through the diffuser without any flow separation and uniform exit velocity profile with exit velocity of 50.48 m/s at outer annulus side,50.14 m/s at inner annulus side which justifies the diffuser design as shown in optimal diffuser configuration in Figure 7.

**Velocity contour for different geometry**

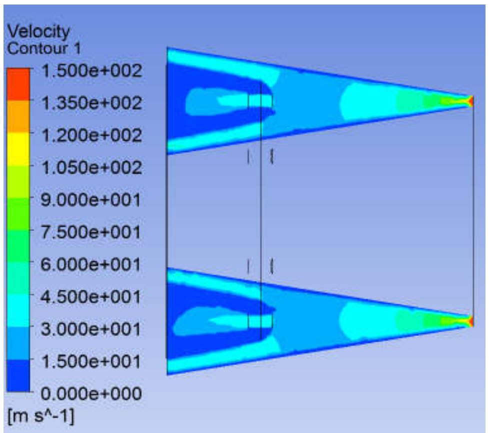


**Figure 3 Hole Dia=5.34 mm**

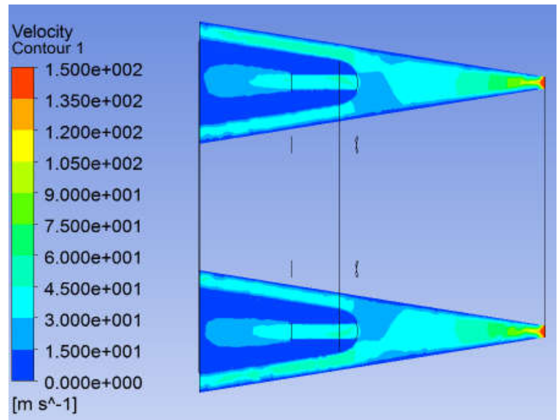


**Figure 4 Hole Dia =5.6 mm**

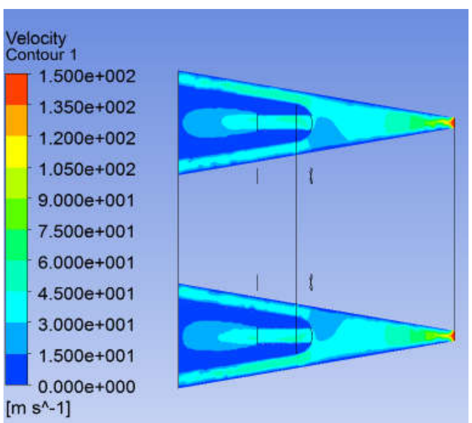


**Figure 5 Hole Dia = 6.2 mm**

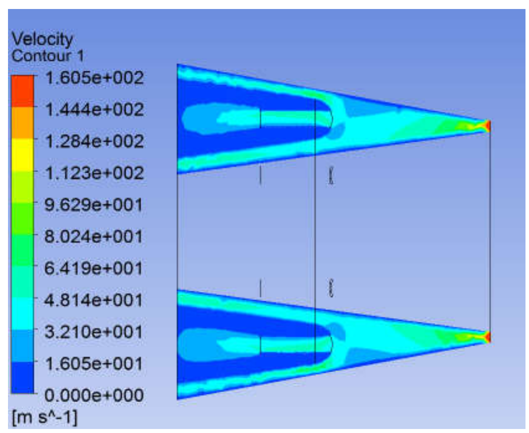


**Figure 6 Hole Dia = 6.5 mm,inner radius 36 mm**

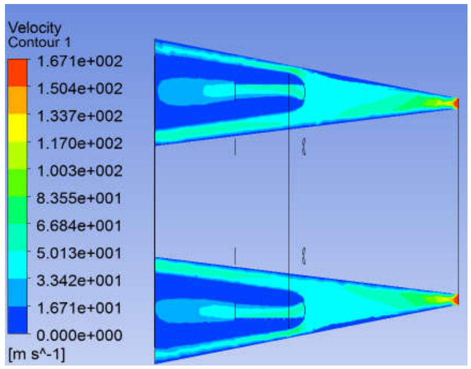


**Figure 7 Hole Dia = 6.5 mm,inner radius 34.5 mm**

Mass flow rate obtained in primary design and optimized design is shown in below table, which shows that after optimization of preliminary design desire mass flow rate achieve with maximum variation of 6.41% for snout outlet which is acceptable result.

Table 3: mass flow rate of primary design and optimized design

| | Hole Diameter (mm) | Inner radius (mm) | Outer annulus mass flow rate (kg/s) | Inner annulus mass flow rate (kg/s) | mass flow rate (kg/s) |
|---|---|---|---|---|---|
| Primary design | 5.34 | 38.65 | 0.2272 | 0.09897 | 0.01124 |
| Optimized design | 6.2 | 34.5 | 0.1874 | 0.1339 | 0.01652 |

Final physical dimension for optimal configuration of diffuser are selected as per below listed table.

Table 4: Final physical dimension of diffuser

| | Mass flow rate (kg/s) | Velocity (m/s) | Outer radius (mm) | Inner radius (mm) | Length (mm) | Hole dia (mm) |
|---|---|---|---|---|---|---|
| Diffuser inlet | 0.3377 | 150 | 42.19 | 34.50 | 11.27 | 3.1 |
| Snout inlet | 0.01546 | 50 | 40.83 | 40.30 | 34.71 | 3.1 |

## V. CONCLUSIONS

From the CFD simulation of designed diffuser it is found that for preliminary design desire mass flow rate is not achieved through the inner annulus and snout outlet further numerical optimization is carried out by changing hole diameter and inner radius of diffuser inlet and result shows that desire mass flowrate and velocity is achieved without any flow separation when inner radius is 34.5 mm and hole diameter is 6.2 mm which justify the design of diffuser.

## REFERENCES


[1] Adkins R. C. and Wardle M. H, *A method for the design of Optimum Annular Diffuser*, Journal of Engineering for Gas Turbine and Power **114** (1990).

[2] Digvijay Kulshreshtha, Dr. S. A. Channiwala, and Saurabh Dikshit, *Numerical Simulation to Study the Effect of Divergence Angles on Performance of Inlet Diffuser Section of a Gas Turbine Combustor*, 2nd International Congress on Computational Mechanics and Simulation (2006).

[3] S. J. Kline, D. E. Abbott, and R. W. Fox, *Optimum Design of Straight-Walled Diffusers*, J. of Basic Eng. **81** (1959) 321-331.

[4] G. Sovran and E. D. Klomp, *Experimentally Determined Optimum Geometries for Rectilinear Diffuser with Rectangular, Conical or Annular Cross- section,* Fluid Mechanics of Internal Flow, Elsevier, New York, (1967) 270-319.

[5] J. H. G Howard, A. B. Thorton-trump, and H. J. Henseler, *Performance and Flow Regimes for Annular Diffusers*, ASME (1967), Paper No.: 67-WA/FE-21 (A68-11861).

[6] H. Peters, *Conversion of Energy in Cross-Sectional Divergences*, 1934.

[7] Lefebvre A H, Gas Turbine Combustion Vol.3, ISBN: 978-1-4200-8605-8, 3rd edition, CRC press, Taylor and fransis group, 2010.

[8] Mellor A. M., Design of Modern Turbine Combustors*,* Academic Press Inc., 1990.